Autodesk® Revit® 2013
Autodesk® BIM 360™

Air, Fluid Flow, and Thermal Simulation of Data Centers with Autodesk Revit 2013 and Autodesk BIM 360

Data centers consume approximately 200 terawatt hours of energy worldwide and were estimated to account for almost 2 percent of total U.S. energy consumption in 2010.[1] Constructing new data centers and retrofitting existing sites to be more energy efficient is a major priority in the industry. This paper will discuss how using computational fluid dynamics cloud-based services within the Autodesk 360[2] platform as part of the BIM process enables MEP engineers using Autodesk Revit[3] software to move computational-heavy simulation tasks to the cloud, helping them to better predict, optimize, and validate their designs early in the design process.

Mechanical, electrical, and plumbing (MEP) engineers can drive cost benefits to their clients by using energy conservation measures (ECMs) as a thermal management strategy on data centers. By performing computational fluid dynamic simulations to their Autodesk® Revit® design models, engineers can make more informed decisions on the systems to help optimize operation and maintenance costs for data center owners.

Whereas direct liquid cooling, direct generation (DG), and combined heat and power (CHP) have not been widely adopted by data center designers due to cost and risk, systematic implementation of energy conservation measures is an increasingly common thermal management strategy. ECMs frequently result in low-complexity/high-yield modifications that lead to significant energy and cost savings.

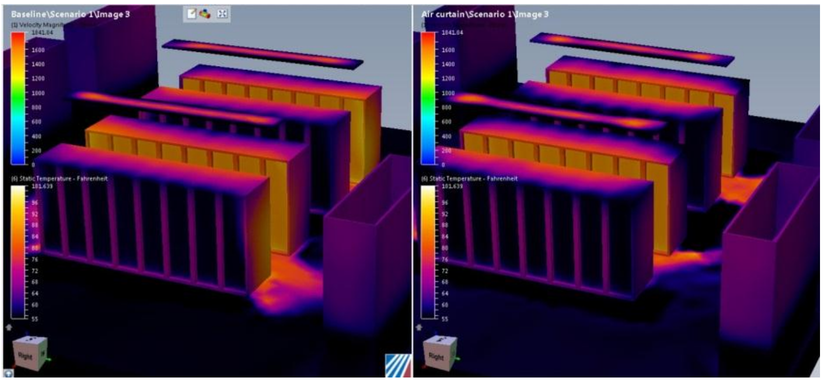
[1] Data-Center Green Movement Gains Ground. Engineering News-Record. Issue: 09/10/2012.

[2] Autodesk BIM 360 air, fluid flow, and thermal simulation requires Subscription to Autodesk Simulation 360.

[3] Autodesk Revit 2013 contains the functionality of Autodesk® Revit® Architecture 2013, Autodesk® Revit® MEP 2013, and Autodesk® Revit® Structure 2013 software, and is available in the Autodesk® Building Design Suite 2013 Premium and Ultimate editions. To limit the use of product name repetition, the use of the name “Revit” throughout this paper refers to both Autodesk Revit and Autodesk Revit MEP.

Autodesk®

Computational fluid dynamics (CFD) contributes to ECMs by helping designers examine and improve airflow management. CFD analysis can be performed on data centers to examine hot aisle/cold aisle arrangements, intelligent placement of floor tiles, and baffle curtains or blanking panels to prevent improper mixing and exhaust recirculation.

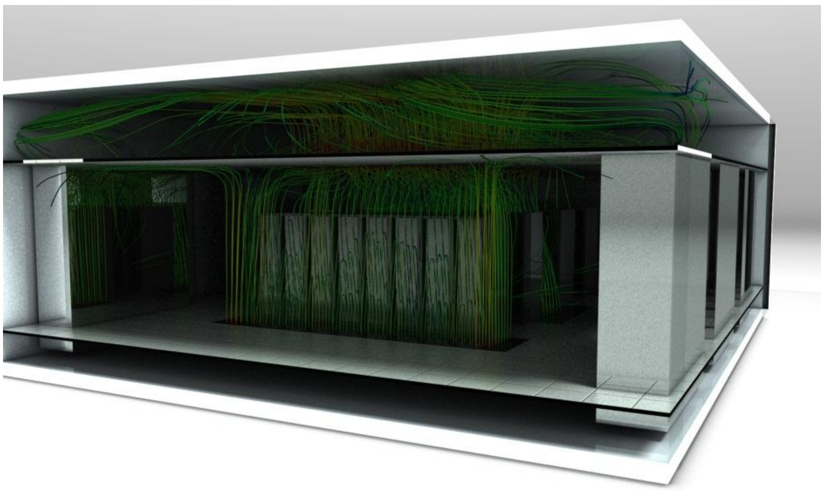


Retrofits or careful placement of CRAC (computer room air conditioner) or CRAH (computer room air handler) units can also deliver major benefits.

Insights early in the design process

With help from Revit and computational fluid dynamics, data center designers can simulate the airflow and thermal response for new designs well before ground is broken at the site. CFD supports visualization of exhaust flow recirculation, understanding of pressure and flows in the subfloor, and quick identification of areas where cool air bypasses server racks. These insights enable designers to more quickly optimize tile configurations, evaluate the impact of new CRAC units or subfloor baffles, and experiment with failure scenarios or rack-load cycling.

Common analysis targets include pressure distributions, rack inlet and exhaust temperatures, CRAC set points and return temperatures, vectors, tracers, and even transient thermal responses when losing CRAC units.



A near unlimited number of virtual thermocouples and rack monitors can be included in the analysis. Once the baseline is characterized and outputs are defined, simulations examining other design options can be set up within minutes. By helping MEP engineers to investigate multiple design strategies to help converge on the optimal solution, designers can provide more effective and efficient scenarios to their clients.

For retrofits, the MEP engineer can use existing drawings or site measurements to model the existing data center, characterize performance, and then implement ECMs virtually to help determine energy savings, break-even points, and year-over-year cost savings once the changes are implemented.

A real-world air, fluid flow, and thermal simulation example

The following images and scenarios represent a real-world example and results from using air, fluid flow, and thermal simulation in a data center environment. Autodesk® Simulation CFD software was used to perform air, fluid flow, and thermal simulation on the proposed layout of a data center, as well as on two modified configurations of the same data center. Flow and heat transfer phenomenon were studied:

- from the CRAC units
- in the subfloor region
- through the floor tiles in the code aisles
- around and through the cabinets and other obstructions
- through the ceiling tiles in the host aisles
- around the plenum in the ceiling of the data center
- back into the CRAC units

Particular attention was paid to the heat distribution across the rear of the cabinets in the hot aisles, with the goal of attaining even temperature distribution across the backs of all cabinets, and to determine if temperatures exceeded server specifications.

Using Revit, three-dimensional models were created and used for analysis within Autodesk Simulation solutions. These models provided more accurate representations of: the cabinets; combined server thermal and fan performance; ceiling and floor tiles of various free area ratios; internal fencing; CRAC units; subfloor; and above-ceiling plenums.

Simulation assumptions

Certain assumptions were made for these simulations:

- Steady state conditions
- Incompressible flow
- Standard atmospheric conditions
- Constant flow rates for all blowers and fans, including servers and CRAC units
- Constant 55° F air supplied by CRAC units
- Even distribution of heat and airflow within each cabinet
- No heat loss through walls, floors, or ceiling

Simulation setup

Cabinet data was based on the Energy Star Power and Performance data sheet for the Dell PowerEdge R610 with high-output 717W power supply. The maximum heat load condition of 425W per unit was assumed, as was the nominal airflow (26 cfm) at nominal ambient air temperature (65° F to 80° F). This constitutes a conservative performance scenario, as opposed to using lower heat loads and/or higher fan rates at higher nominal air temperatures. The data center configuration was based on the assumption that 14kW cabinets would hold 30 servers per cabinet, 7kW cabinets held 15 servers per cabinet, and 2.5kW cabinets held 7 servers per cabinet.

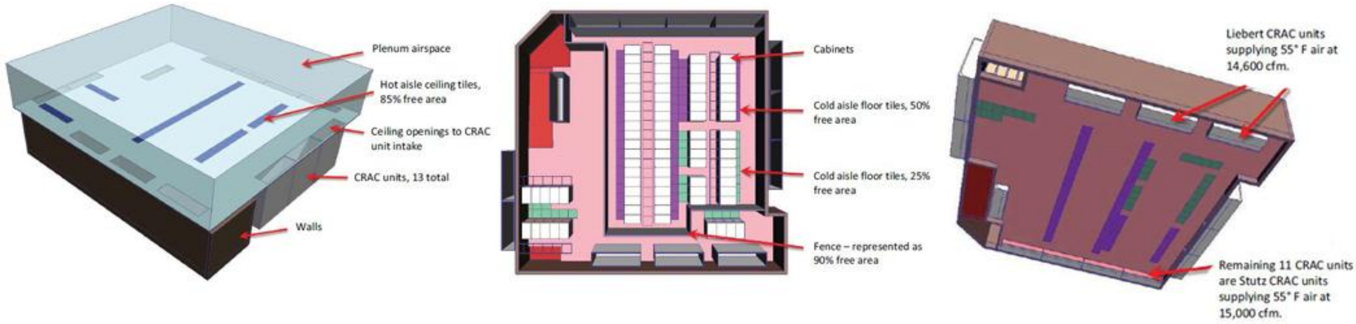


Figure 1. Model views ready for simulation setup.

Baseline results

Examining the temperatures across the back planes of the cabinets shows a mostly uniform temperature distribution across all cabinets. There are areas, however, where there are higher temperature concentrations. These areas are at the ends of the hot aisles and can be seen below. The temperature rise from inlet to exhaust of all other cabinets is approximately 86° F, in keeping with the Energy document provided by Dell for these systems.

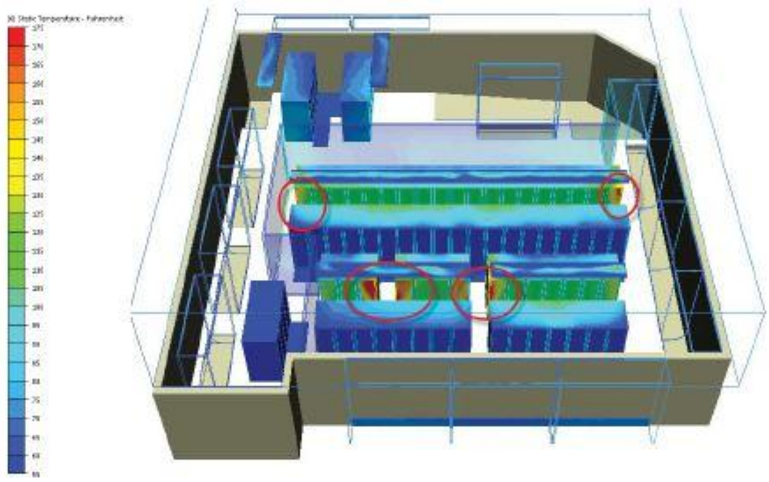


Figure 2. Localized increase in cabinet exhaust temperatures at ends of hot aisles.

Further examination of the flow fields showed that hot exhaust air was escaping from around the corners of the cabinets therefore driving up local temperatures.

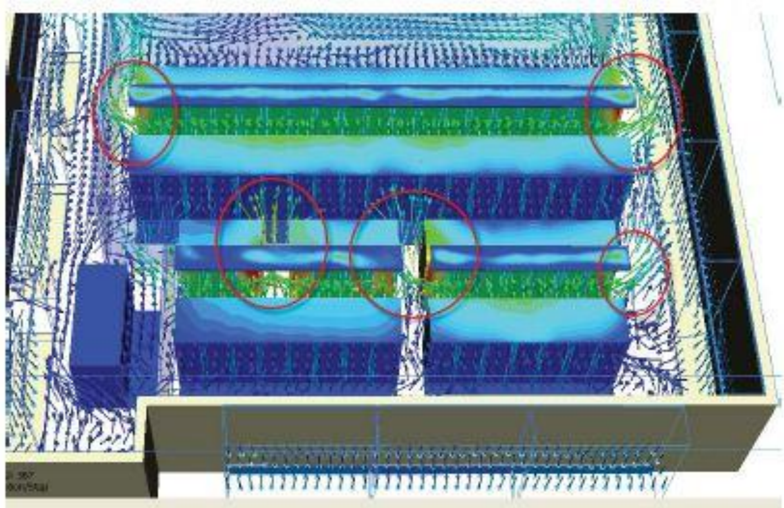


Figure 3. Closer view of exhaust air escape patterns.

An air curtain was created in hopes of containing the escaping hot aisle air and thereby reducing the temperature concentrations on cabinets at the end of the aisles. While all other setup conditions were unchanged, additional floor and ceiling tiles were added to create an air curtain effect at openings in the hot aisle.

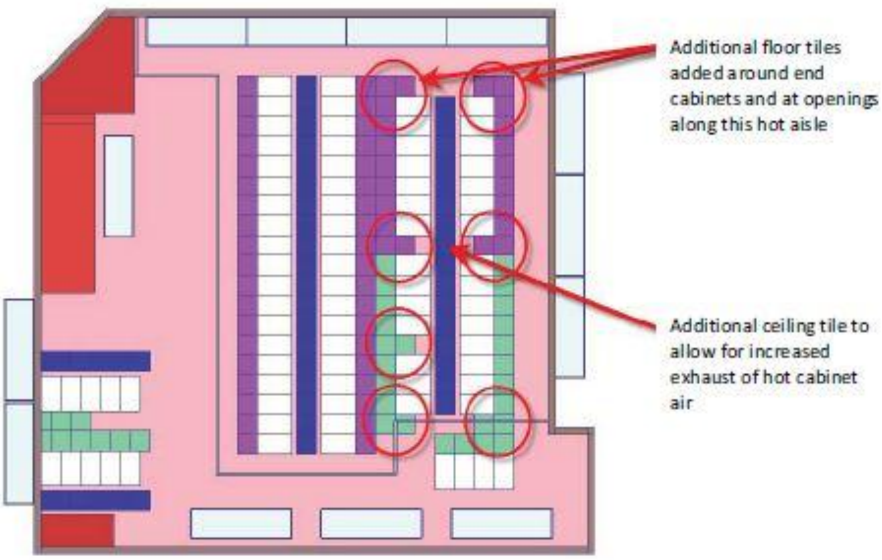


Figure 4. Air curtain model setup. Plan View – Main Floor.

The hot aisle with the added air curtain feature shows a notable reduction of the high temperature concentrations at the end of the modified aisle.

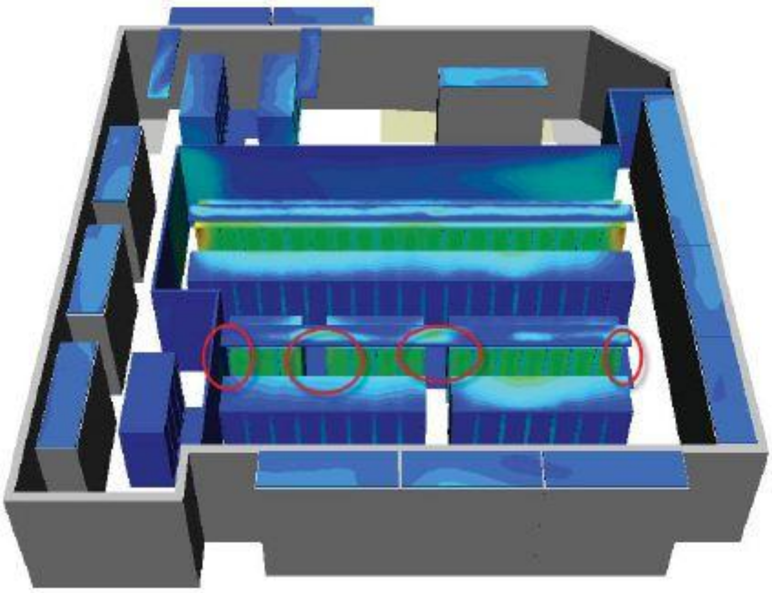


Figure 5. Equal cabinet exhaust temperatures at ends of hot aisle with air curtain

Reviewing the flow fields in these areas reveals that hot exhaust air from the cabinets is now contained around these corners, providing uniform temperatures across all units in this aisle.

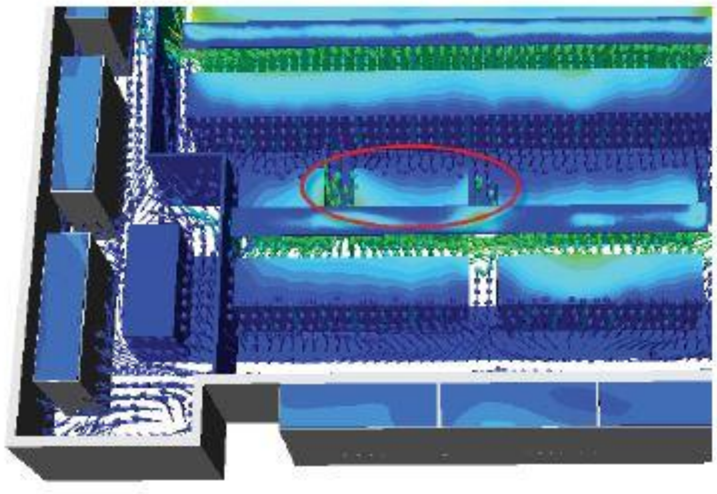


Figure 6. Cabinet exhaust air contained in hot aisle. Note more vertical flow than baseline simulation.

Design requirements for the data center changed, requiring more cabinets. Rearranging the location of the higher-density cabinets also required a different floor tile distribution, as shown below. All other setup information is the same as for the air curtain simulation. The additional cabinets are still within the max temperature gain expressed by Dell for the individual systems, implying that there is adequate cooling for the additional systems. Temperature distribution is still even across the backs of the cabinets in the hot aisle contained with the air curtain. In contrast, the new row of cabinets is showing signs of temperature increase near the ends of the aisles.

Figure 7. Aerial model view of cabinet exhaust air contained in hot aisle.

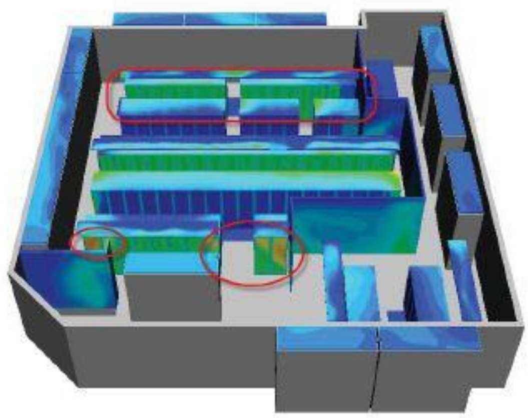


Figure 8. Additional cabinet configuration showing temperature increase.

Conclusions

Airflow in the subfloor plenum appears evenly distributed so as to provide even cooling to the entire data center across a range of cabinet floor/ceiling tile configurations. Tile opening sizes and locations appear adequate for providing sufficient cooling for each cabinet and for evenly distributing the cooling among cabinets of different densities. However, the addition of partially open floor and ceiling tiles around gaps in the aisles and at the ends of the aisles may provide better containment of hot aisle exhaust, and therefore better cooling of cabinets at the ends of aisles and near gaps in the cabinet spacing. Finally, there appears to be adequate cooling for additional cabinets.

Autodesk Revit software and air, fluid flow, and thermal simulation enabled a better understanding of air, fluid flow, and thermal simulations and how those aspects would affect the overall operation of the data center. These insights enabled the MEP engineers to make better design decisions before construction took place, helping to provide an optimum scenario to the owner.